Watershed Modeling System

WMS v8.4

TUTORIALS – Volume 4

STORM DRAIN MODELING

WMS 8.4 Tutorials – Volume 4

Copyright © 2010 Aquaveo, LLC

All Rights Reserved

Unauthorized duplication of the *WMS* software or documentation is strictly prohibited.

AQUAVEO LLC MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *WMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *WMS* is a product of the Aquaveo, LLC. For more information about this software and related products, contact Aquaveo at:

Aquaveo
75 South 200 East, Suite 201
Provo, Utah 84606
Tel.: (801) 691-5528
e-mail: wms@aquaveo.com
WWW: http://www.aquaveo.com/

For technical support, contact Aquaveo's tech support number at (801) 691-5530 (Monday-Friday, 8am-5pm Mountain Time)

TABLE OF CONTENTS

1 SWMM MODELING 1-1

1.1 OBJECTIVES 1-1
1.2 DEFINING RUNOFF COEFFICIENTS 1-2
1.3 COMPUTE FLOWS FROM THE RATIONAL METHOD 1-3
1.4 IMPORTING THE PIPE NETWORK 1-7
1.5 RUNNING XPSWMM AND VIEWING RESULTS 1-11
1.6 RUNNING EPA SWMM AND VIEWING RESULTS 1-13
1.7 ROUTING HYDROGRAPHS IN SWMM 1-17

2 STORM DRAIN: RATIONAL DESIGN 2-1

2.1 OBJECTIVES 2-1
2.2 DEFINING RUNOFF COEFFICIENTS 2-2
2.3 DEFINING THE DRAINAGE AREA 2-3
2.4 IMPORTING THE PIPE NETWORK 2-8
2.5 LINKING NODES AND ASSIGNING ELEVATIONS 2-13
2.6 SAVING THE SIMULATION AND RUNNING STORM DRAIN 2-14

3 STORM DRAIN: HYDROGRAPHIC DESIGN 3-1

3.1 OBJECTIVES 3-2
3.2 DEVELOPING THE SURFACE DRAINAGE COVERAGE 3-2
3.3 RUNNING A RATIONAL ANALYSIS 3-6
3.4 CREATING THE PIPE NETWORK 3-8
3.5 SAVING THE SIMULATION AND RUNNING STORM DRAIN 3-13

CHAPTER 1

SWMM Modeling

SWMM is a hydrologic and hydraulic analysis tool used primarily designing and analyzing for storm drain systems. Hydrologic parameters are defined to obtain hydrographs and peak flows which can then be fed into the hydraulic component of the model. The hydraulic component lets engineers analyze the capacity of a current storm drain system or design a system to meet certain inflow conditions. WMS supports both EPA SWMM, developed by the Water Supply and Water Resources Division of the US Environmental Protection Agency's National Risk Management Research Laboratory, and xpswmm, developed by XP Software.

1.1 Objectives

In this exercise, we will set up a drainage simulation based on the Rational Method for a proposed subdivision. The objective of this exercise is to teach you the basic steps for defining a SWMM input file, running the numeric model, and viewing the results. These steps include the following:

1. Define runoff coefficients
2. Compute flows using the Rational Method
3. Import a pipe network
4. Run SWMM and view results
5. Route hydrographs using SWMM and view results

1.2 Defining Runoff Coefficients

1.2.1 Reading in the Subdivision Layout and Elevations

We will open a shapefile containing the geography of the proposed subdivision. Also, to help us in defining elevations and slopes for the drainage area, we will open a TIN (Triangulated Irregular Network) for the area:

1. Close all instances of WMS
2. Open WMS
3. Select ***File*** | ***Open***
4. Locate the folder ***C:\Program Files\WMS84\tutorial\stormrat***
5. Open *"cougarestates.tin"*
6. Right-click on the Coverages folder in the Project Explorer
7. Select ***New Coverage*** from the pop-up menu
8. Change the Coverage type to *Runoff Coefficient*
9. Select *OK*
10. Select ***File*** | ***Open***
11. Open *"cougarestates.shp"*
12. Switch to the *GIS* module
13. Select ***Mapping*** | ***Shapes->Feature Objects***
14. Select *Yes* to use all visible shapes
15. Select *Next*
16. Ensure that the Runoff coefficient is mapped
17. Select *Next* and then *Finish*
18. Hide *cougarestates.shp* by deselecting it in the Project Explorer

1.2.2 Defining the Runoff Coefficients

We will turn off the display of the TIN in order to better distinguish the subdivision polygons.

1. Toggle the check box for the "new" TIN off in the Project Explorer
2. Switch to the *Map* module
3. Choose the *Select Feature Polygon* tool
4. Use Figure 1-1 to assign runoff coefficients by double clicking each polygon and assigning the appropriate coefficient.

Figure 1-1: Assigning runoff coefficients.

1.3 Compute Flows from the Rational Method

Now we will set up our Rational Method analysis to compute peak flows for each subbasin. We will use the computed peak flows in our SWMM storm drain network.

1.3.1 Creating the Drainage Coverage

1. Select the *Drainage* coverage from the Project Explorer to make it the active coverage
2. Select ***File | Open***
3. Open *"runoff.shp"* and *"drainage.shp"*
4. Switch to the *GIS* module

5. Select ***Mapping | Shapes->Feature Objects***

6. Select *Yes* to use all visible shapes

7. Select *Next*, *Next*, *Next*, and then *Finish*

Figure 1-2 shows how the layout should look.

8. Hide *runoff.shp* and *drainage.shp* by deselecting their icons in the Project Explorer

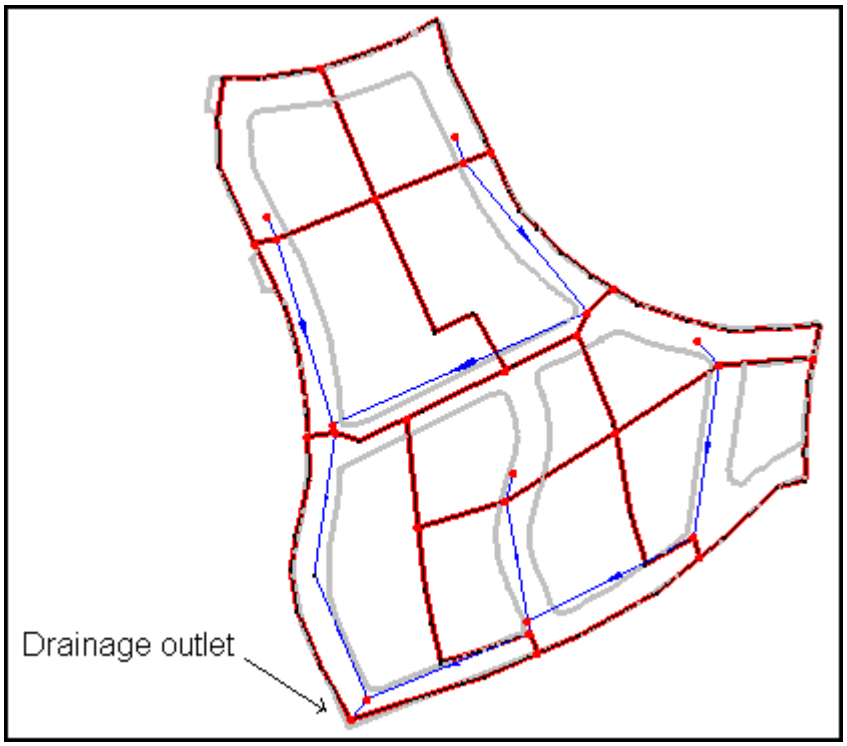


Figure 1-2: Drainage basins and streams for the subdivision.

Before defining nodes as storm drains/drainage outlets, we will hide the Runoff Coefficient coverage in order to simplify the screen display

9. Toggle the visibility check box for the *Runoff Coefficient* coverage off in the Project Explorer

10. Ensure that *Drainage* is the active coverage and that your current module is the *Map* module

11. Choose the *Select Feature Point/Node* tool

12. Double-click on the node labeled Drainage Outlet in Figure 1-2

13. Change the Point Type to *Drainage Outlet* and select *OK*

14. Select ***Feature Objects | Compute Basin Data***

15. Click the *Current Coordinates* button

16. Set the horizontal system to use a local projection and set the horizontal and vertical units as *U.S. Survey Feet*.

17. Select *OK*

18. Set Basin Areas to *Acres* and Distances to *Feet*

19. Select *OK*

1.3.2 Computing Runoff Coefficients

Composite runoff coefficients must be computed for each drainage area. A weighted average for each drainage area is calculated from data in the Runoff Coefficient coverage. To compute the basin runoff coefficients:

1. Switch to the *Hydrologic Modeling* module

2. Select ***Calculators | Compute GIS Attributes***

3. Set the Computation type as *Runoff coefficients*

4. Select *OK*

Composite runoff coefficients for each drainage area are computed and displayed on the screen.

1.3.3 Entering Times of Concentration

The Time of Concentration for a basin can be calculated using a Time Computation coverage. For this exercise, however, it is assumed that the TC values have already been calculated, and we will input them manually.

1. Change the Model drop-down box (located towards the top of the WMS interface) to *Rational*

2. Choose the *Select Basin* tool

3. Double-click on the basin icon for the basin in the lower left-hand corner of the subdivision

4. Enter **14** (minutes) for the Time of Concentration

5. Select each of the remaining drainage basins and assign Time of Concentration values using Figure 1-3 as a guide

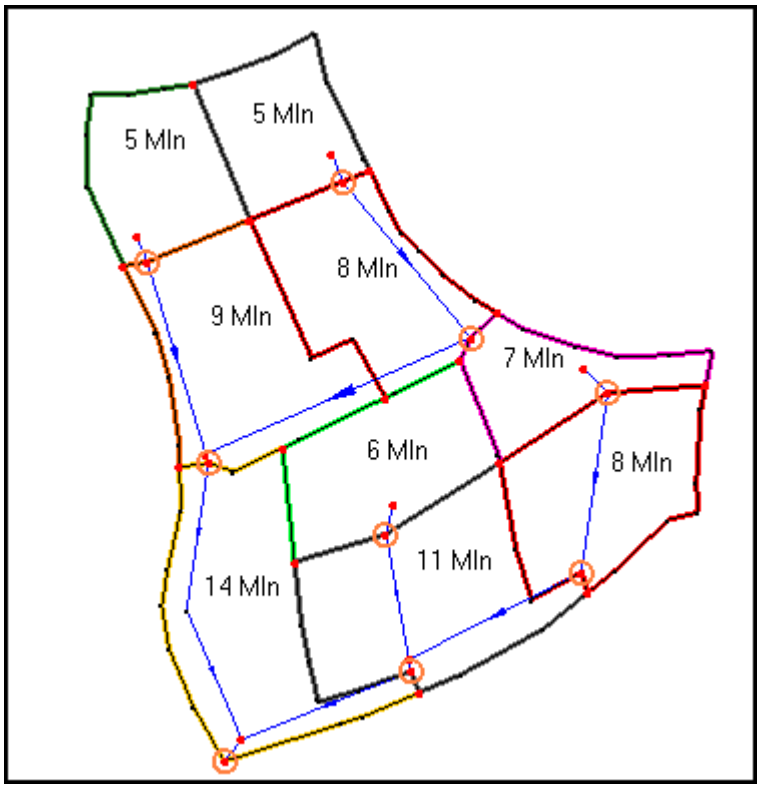


Figure 1-3: TC values for the drainage areas.

6. Select *OK* to close the Rational Method dialog

We have now set up a traditional rational simulation for the Cougar Estates subdivision. The only remaining step is to define the IDF curves.

1.3.4 Defining Rainfall Data

1. Double click on the basin icon for the basin in the lower left hand corner of the subdivision

2. Click on the *Compute…* button for the parameter labeled *Compute I – IDF Curves*

3. Choose the *User Supplied Data* option as the IDF curve computation

4. Click the *Define Storm Data* button

5. Change the Recurrence value to **25** yr

6. Enter the precipitation values shown in Figure 1-4

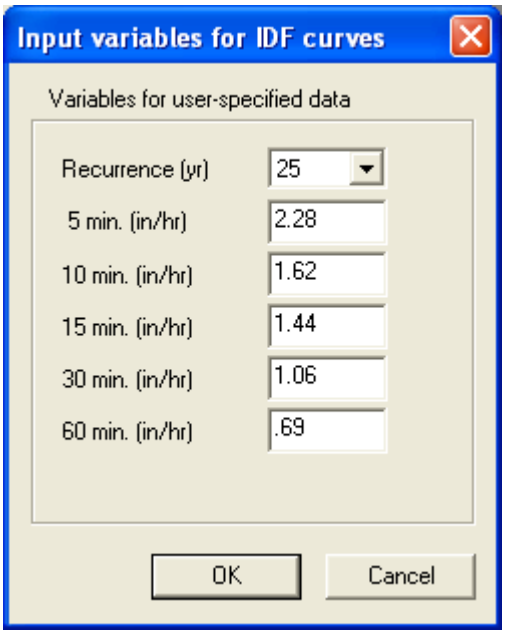


Figure 1-4: Values for computing the 25 yr IDF curve.

7. Select *OK*

8. Highlight the line corresponding to the 25-yr precipitation values from the window in the upper right-hand corner of the dialog

9. Click the *Compute Intensity* button.

10. Select *Done*

11. In the *Display* section, change the *Type* drop down menu to *Outlets* and change the *Show* drop down menu to *All*

12. Click the *Compute…* button next to the *Compute I – IDF Curves* parameter for each basin and outlet. Click the *Compute Intensity* button and click *Done* until each basin and outlet has a rainfall intensity displayed in the *Parameters* spreadsheet

13. Notice that a flow rate has now been computed for each basin and outlet. Click *OK* to exit the Rational Method dialog

We will now import a pre-defined storm drain network, assign properties to the network, and link it to the Drainage coverage.

1.4 Importing the Pipe Network

As mentioned above, we will be importing a pipe network to use in our SWMM model. This network was defined as a shapefile, and will be converted

to feature objects in WMS. WMS can also open DXF files and convert them to feature objects. Alternatively, users can manually create a pipe network using the Create Feature Arc tool.

1. Right-click on the Coverages folder in the Project Explorer
2. Select ***New Coverage***
3. Change the Coverage type to *Storm Drain*
4. Select *OK*
5. Select ***File | Open***
6. Open *"swmm_pipes.shp"*
7. Switch to the *GIS* module
8. Select ***Mapping | Shapes->Feature Objects***
9. Select *Yes* to use all visible shapes
10. Select *Next*, *Next*, and *Finish*
11. Hide *swmm_pipes.shp*

1.4.1 Apply SWMM Attributes

1. Switch to the Map module
2. Select ***Storm Drain | Map->1D Schematic***

This will convert our storm drain coverage to a network of links and nodes recognized by SWMM. This command also automatically maps the elevations from our TIN to the ground elevations stored at each node. WMS has also computed the length of each storm drain link and applied a default invert elevation of eight feet below the ground elevation. Your project should now look similar to Figure 1-5.

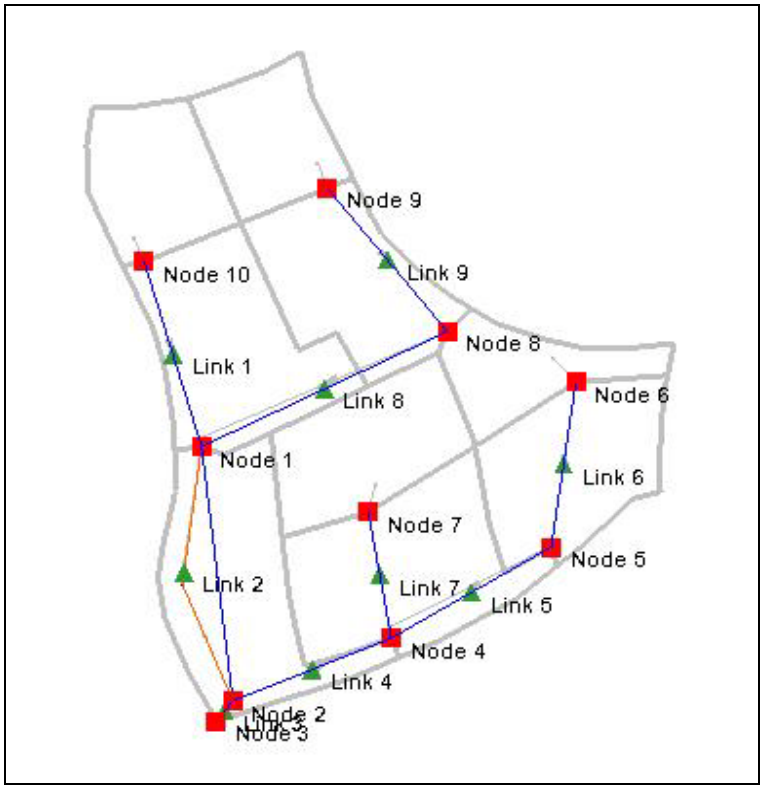


Figure 1-5 Link-Node Schematic for SWMM Model

3. In the Project Explorer, select the *SWMM Hydraulic Schematic* tree item

4. Choose the *Select Hydraulic Node* tool

5. Double click on any node

In the node Properties dialog, notice the invert elevation of Node 10 and Node 1 (according to Figure 1-5). Storm water is supposed to flow from Node 10 to Node 1, but the default invert elevation is higher at Node 1 than at Node 10. We have a similar issue with respect to Node 9 and Node 8. This dialog lets us change the default invert elevations at each node.

Important Note: for the following steps, node numbers may not match node numbers in your model. Use Figure 1-5 as a reference to determine which nodes correspond to node and link names in your model.

6. Change the invert elevation of Node 1 to **4501.8**

This elevation must be higher than that at Node 2. The elevation at Node 2 is 4501.5, so we've met this condition

7. Change the invert elevation of Node 9 to **4508.0**

This elevation is now higher than the elevation at Node 8 (4507.5). Now notice the column labeled Linked Outlet Name. The storm drain nodes are not currently linked to our hydrologic analysis. We want to use the peak flows calculated in our Rational Method Analysis as inflows to our storm drain system.

8. Click *OK* on the node Properties dialog

9. Right click on the *SWMM Hydraulic Schematic* in the Project Explorer and select *Link Outlets to Nodes*

10. In the Link Storm Drain and Drainage Nodes dialog, click the *Auto Link* button.

This will automatically link the storm drain nodes to the drainage outlets that are within a given tolerance of the node. As you can see, Node 2 is not located close enough to any drainage outlet and so has not been linked

11. Click *OK* to exit the Link Storm Drain and Drainage Nodes dialog

Now that we've changed the elevations at our nodes and linked them to our outlets, we must also modify the upstream and downstream invert elevations of our links.

12. Choose the *Select Hydraulic Link* tool

13. Double click on any link

The link Properties dialog lets us specify the dimensions of our storm drain pipes. We'll assume a circular shape for all pipes and no initial flow or depth. The lengths and invert elevations are shown, but the invert elevations at Link 1 and Link 9 need to be updated to reflect the changes we made to the nodes. We also need to define the diameter of the pipes. **Remember:** Use Figure 1-5 as a reference to determine which links correspond to link names in your model.

14. Change the Downstream Invert Elevation of Link 1 and the Upstream Invert Elevation of Link 2 to **4501.8**

15. Change the Upstream Invert Elevation of Link 9 to **4508.0**

16. In the *Diameter/Height* column of the link properties spreadsheet, enter a diameter of **4** feet for each link

17. Click *OK* to exit the link Properties dialog

Now we're ready to export a SWMM project file and run SWMM. Section 1.5 describes how to export and run an xpswmm model. To export and run an EPA SWMM model, skip to Section 1.6.

1.5 Running xpSWMM and Viewing Results

This section describes how to export and run an xpswmm model. The xpswmm model offers many features that can be used to modify your hydrologic analysis, customize your drain system, and analyze the model output. The features covered here represent just a few of the features you might find helpful.

1. Select ***SWMM | Run xpswmm…***

2. Name the file "*swmmtut.xpx*" and click OK

WMS saves an xpswmm project file and as long as xpswmm is loaded on your computer, it launches xpswmm and automatically opens the project we just created.

1.5.1 Running the SWMM Model

1. When xpswmm opens, if prompted click *OK* to convert the version 10.0 database and close the window displaying errors and warnings.

2. Your xpswmm window should now display an image similar to that shown in Figure 1-6.

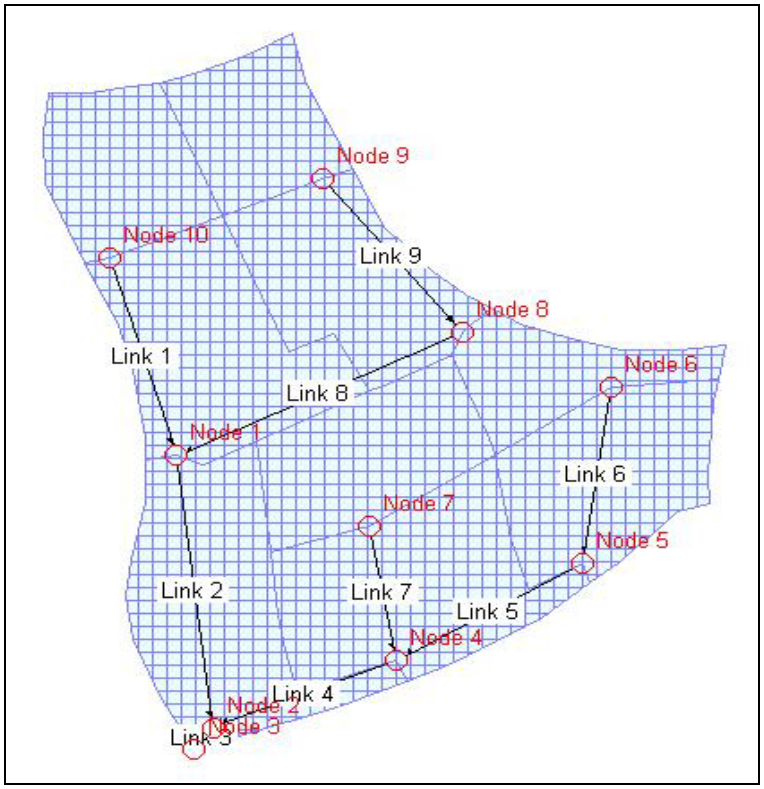


Figure 1-6 Cougar Estates Storm Drain Schematic from xpswmm

3. Double click on the node labeled “Node 9” in Figure 1-6

The Node Data dialog displays the ground elevation, invert elevation, and inflow data as well as the ponding type and initial depth if specified in WMS.

4. Click OK to close the Node Data dialog

5. Double click on the link labeled “Link 9” in Figure 1-6

The Conduit Data dialog displays the pipe data we imported from WMS. On the main window, notice the edit fields for initial flow and initial depth.

6. Click on the button labeled *Conduit Profile*

   Here we find information about the pipe dimensions and material as well as the invert and ground elevations for the upstream and downstream ends of the pipe.

7. Click *OK*, then *OK* again to close both dialogs

Now before we can run SWMM, we need to specify an outfall condition at the downstream outlet of our storm drain system and set up the run time parameters.

8. Double click on the most downstream link in the network (Link 3 in Figure 1-6)

9. Click on the *Outfall* button

For this exercise, we’ll assume a free outfall at the downstream outlet.

10. Click on the *Type 1, Free Outfall* button

11. In the Outfall Control dialog, select *Use normal depth (Yn)*

12. Click OK, OK, then OK to exit all the dialogs

13. Select ***Configuration | Job Control | Hydraulics***

14. In the Job Control dialog, change the ending day to **1**, and the ending hour to **3** and click *OK* to exit the dialog

15. Select ***Analyze | Solve*** or click the *Solve* button

16. Name the file “*swmmtut.out*” and click *Save*

1.5.2 Viewing SWMM Output

1. After the model is done running, select any link and select ***Results | Review Results***

This option will open a plot showing the upstream and downstream water surface elevations at any point throughout the model run and the computed flows and velocities.

2. Click *Close* and with the same link selected, select ***Results | Dynamic Section Views***

3. Click the *Play* button

This option will play an animation of the storm flow as it moves through your selected link. It displays both a profile view and cross sectional view of the flow through the pipe and the flow graph we saw in the previous plot. Feel free to explore the rest of the options under the Results menu or close xpswmm. Leave your WMS project open for the next example.

1.6 Running EPA SWMM and Viewing Results

This section describes how to export and run an EPA SWMM model. The EPA SWMM model offers many features that can be used to modify your hydrologic analysis, customize your drain system, and analyze the model output. The features covered here represent just a few of the features you might find helpful.

1. Select ***SWMM | Run EPA SWMM…***

2. Name the file "*epaswmmtut.inp*" and click *Save*

WMS saves an EPA SWMM project file and as long as EPA SWMM is loaded on your computer, it launches EPA SWMM and automatically opens the project we just created.

1.6.1 Viewing SWMM Model Inputs

1. Your EPA SWMM window should display an image similar to that shown in Figure 1-7.

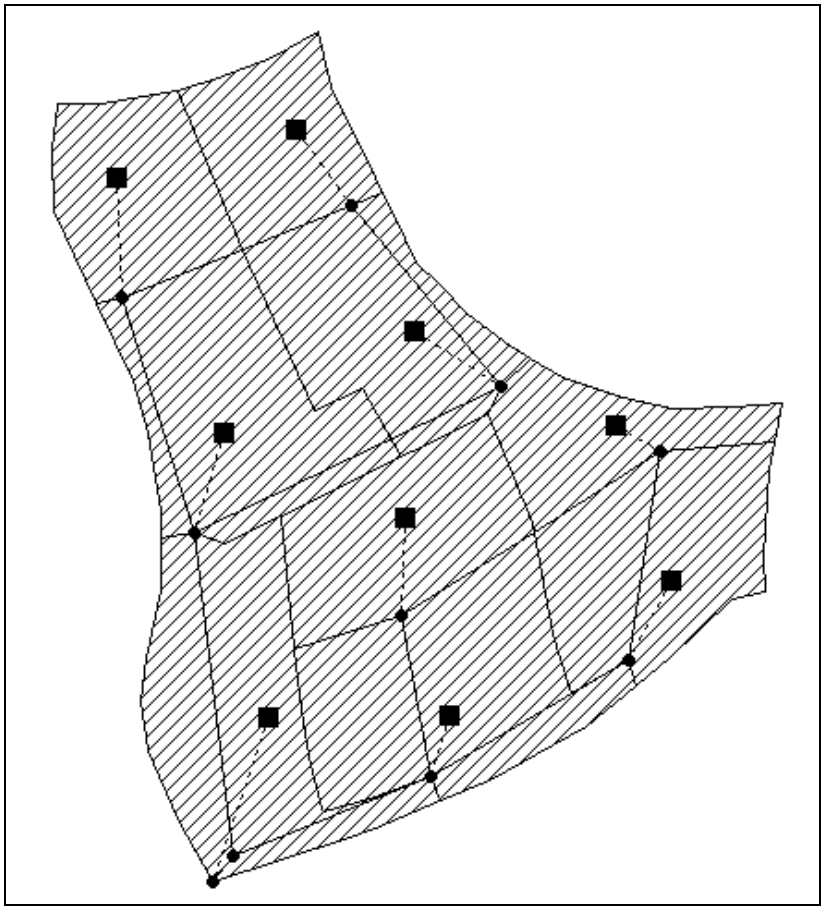


Figure 1-7 Cougar Estates Storm Drain Schematic from EPA SWMM

2. In the SWMM Data Explorer, expand the Hydraulics heading, then the Links heading, and select Conduits

3. In the Conduits window below, select one of the Links

4. Double click the blinking link icon corresponding to the link you selected

5. In the Conduit Link Properties window, notice the link shape, maximum depth, and length which were specified and computed in WMS and exported to EPA SWMM

6. Close the Conduit Link Properties window

7. In the SWMM Data Explorer, expand the Hydraulics heading, then the Nodes heading, and select Junctions

8. In the Junctions window below, select a Node

9. Double click on the blinking node icon corresponding to the node you selected

10. In the Junction Node Properties window, notice the Invert Elevation of the node which was computed in WMS and exported to EPA SWMM

11. Close the Junction Node Properties window

12. In the SWMM Data Explorer, expand the Hydrology heading and select Subcatchments

13. In the Subcatchments window below, select a subcatchment

14. Double click on the blinking subcatchment icon corresponding to the subcatchment you selected

15. In the Subcatchment Properties window, notice that the subcatchment area has already been computed and imported from WMS

Feel free to explore the rest of the model inputs within the EPA SWMM interface.

1.6.2 Running the SWMM Model

1. On the EPA SWMM menu, select ***Project | Run Simulation***

In the Run Status window, you'll notice that the run was successful, but that there was no surface runoff or flow routing.

2. On the Run Status window, click *OK*

1.6.3 Viewing SWMM Output

1. Select ***Report | Status***

   If you scroll down to the subcatchment report you'll notice that all the precipitation was lost to infiltration. Let's take a closer look at our basin parameters to see why there was no runoff.

2. Close the Status Report

3. Double click on a subcatchment to open the Subcatchment Properties dialog

4. Notice that the Width, % Slope, and % Imperv fields are all zero

The width of overland flow path and the % Impervious must be specified inside the EPA SWMM interface. The % Slope is written by WMS, but only when elevations are read from a DEM. Since this tutorial used a TIN instead of a DEM, no basin slopes could be computed.

5. Enter a Width of **2** (feet), a % Slope of **0.02**, and a % Imperv of **80**

6. Repeat step 5 for all Subcatchments
7. Select ***File*** | ***Save***
8. Select ***Project*** | ***Run Simulation***
9. On the Run Status window, click *OK*
10. Open the Status report and notice the runoff values for each subcatchment
11. Close the Status Report
12. Select the conduit connecting the most downstream outlet to the junction just upstream
13. Choose the *Time Series Plot* button
14. Click *OK* on the Time Series Plot window to view the Flow v. Time plot for the most downstream storm drain. The plot should look similar to Figure 1-8

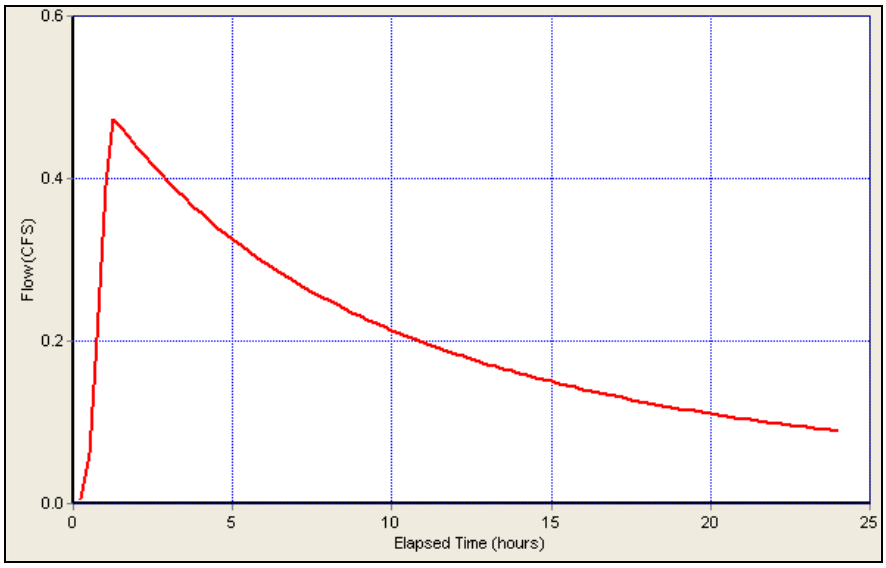


Figure 1-8 Flow v. Time Plot for the most downstream storm drain

Feel free to explore the rest of the EPA SWMM post processing tools to view the results from the Cougar Estates model run. When you are ready to proceed to the next section, close EPA SWMM and leave the WMS window running.

1.7 Routing Hydrographs in SWMM

In the previous example, we used the Rational Method to compute peak flows which we then fed into our SWMM model. This time, we'll use the Rational Method to compute hydrographs at each outlet. We'll then route these hydrographs through our SWMM model.

1.7.1 Computing Rational Method Hydrographs

1. In your WMS project, select the *Drainage* coverage to make it active
2. Switch to the *Hydrologic Modeling* module
3. Choose the *Select Outlet* tool
4. Using the Select Outlet tool, double click on an outlet (it will most likely be hidden behind the SWMM node icon)
5. In Display, change the Show drop down box to Selected
6. Click the *Compute…* button next to the *Compute Hydrographs* parameter for the selected basin and click *Done* in the Rational Method Hydrographs dialog
7. Do the same thing for the selected outlet hydrograph
8. Click *OK* to exit the Rational Method dialog
9. Repeat steps 4 through 7 for all the outlets in the subdivision. By the end, each basin and outlet icon should show a hydrograph icon next to it as in Figure 1-7

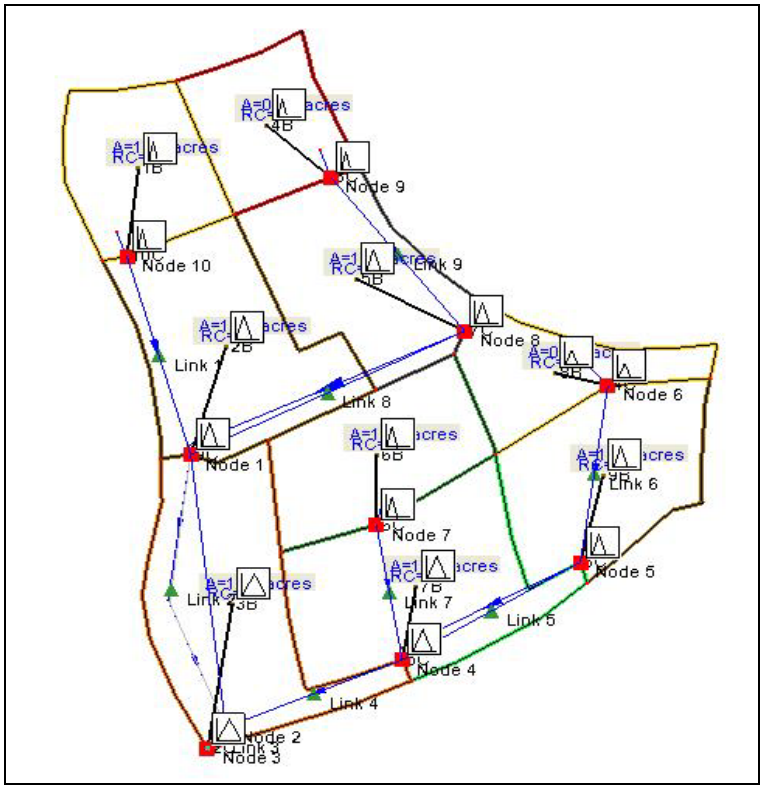


Figure 1-9 Rational Method hydrographs for SWMM

10. In the Project Explorer, select the *SWMM Hydraulic Schematic* tree item

11. Select ***SWMM | Run xpswmm…***

12. Name the file "*swmm_hyd.xpx*" and click OK

1.7.2 Running SWMM and Viewing Results

1. In the xpswmm interface, double click on any node

Notice that the *Inflow* edit field under the Constant Inflow category now says 0.0. This is because we have now defined a Time Series Inflow. Notice how the *User Inflow* button is now checked.

2. Double click the *User Inflow* button to view the hydrograph we will now be routing through the selected node

3. Click OK, then OK to close both dialogs

4. Double click on the most downstream link in the network (Link 3 in Figure 1-6)

5. Click on the *Outfall* button
6. Click on the *Type 1, Free Outfall* button
7. In the Outfall Control dialog, select *Use normal depth (Yn)*
8. Click OK, OK, then OK to exit all the dialogs
9. Select ***Configuration | Job Control | Hydraulics***
10. In the Job Control dialog, change the ending day to **1**, and the ending hour to **3** and then click *OK* to close the dialog
11. Select ***Analyze | Solve*** or click the *Solve* button
12. Name the file *"swmm_hyd.out"* and click *Save*
13. View the results as before and notice the difference between using a constant inflow and a time series inflow

This concludes the SWMM modeling tutorial. Feel free to explore with the WMS interface or the SWMM interface if you wish.

CHAPTER 2

Storm Drain: Rational Design

Storm Drain is a hydraulic analysis and design program for storm drain systems. Developed by the Federal Highway Administration (FHWA), Storm Drain provides hydraulic engineers with a quick and accurate method of computing the capacity of an existing system, or designing a system to meet a given set of input flows. The Storm Drain interface in WMS uses the same HYDRA program that has been distributed by the FHWA as part of their HYDRAIN computer program package for the past several years.

2.1 Objectives

In this exercise, we will set up a drainage simulation based on the Rational Method for a proposed subdivision. The objective of this exercise is to teach you the basic steps for defining a Storm Drain input file, running the numeric model, and viewing the results. These steps include the following:

1. Define runoff coefficients
2. Define the drainage area
3. Compute flows for each basin using the Rational Method
4. Import a pipe network
5. Define the necessary parameters for the network
6. Link nodes from the drainage network to their corresponding nodes in the pipe network

7. Assign elevations to the pipe network
8. Save the Storm Drain input file and run the simulation
9. View the solution

2.2 Defining Runoff Coefficients

2.2.1 Reading in the Subdivision Layout and Elevations

We will open a shapefile containing the geography of the proposed subdivision. Also, to help us in defining elevations and slopes for the drainage area, we will open a TIN (Triangulated Irregular Network) for the area:

1. Close all instances of WMS
2. Open WMS
3. Select ***File | Open***
4. Locate the folder ***C:\Program Files\WMS84\tutorial\stormrat***
5. Open *"cougarestates.tin"*
6. Right-click on the Coverages folder in the Project Explorer
7. Select ***New Coverage*** from the pop-up menu
8. Change the Coverage type to *Runoff Coefficient*
9. Select *OK*
10. Select ***File | Open***
11. Open *"cougarestates.shp"*
12. Switch to the *GIS* module
13. Select ***Mapping | Shapes->Feature Objects***
14. Select *Yes* to use all visible shapes
15. Select *Next*
16. Ensure that the Runoff coefficient is mapped
17. Select *Next* and then *Finish*

18. Hide *cougarestates.shp* by deselecting it in the Project Explorer

2.2.2 Defining the Runoff Coefficients

We will turn off the display of the TIN in order to better distinguish the subdivision polygons.

5. Toggle the check box for the "new" TIN off in the Project Explorer
6. Switch to the *Map* module
7. Choose the *Select Feature Polygon* tool
8. Double-click on the lower left polygon
9. Enter a runoff coefficient of **0.6**
10. Select *OK*
11. Assign Runoff Coefficients for the remaining polygons as shown in Figure 2-1

Figure 2-1: Assigning runoff coefficients.

2.3 Defining the Drainage Area

First, we will initialize the Storm Drain interface. Initializing Storm Drain tells WMS to make available additional features needed to build a Storm Drain

model. Then we will read in pre-defined drainage boundaries and set up drainage inlets and a drainage outfall.

2.3.1 Initializing Storm Drain

1. Change the Model drop-down box located at the top of the WMS interface to *Storm Drain-FHWA*
2. Select ***Storm Drain | New Simulation***

2.3.2 Creating the Drainage Coverage

1. Select the *"Drainage"* coverage in the Project Explorer
2. Select ***File | Open***
3. Open *"runoff.shp"* and *"drainage.shp"*
4. Switch to the *GIS* module
5. Select ***Mapping | Shapes->Feature Objects***
6. Select *Yes* to use all visible shapes
7. Select *Next*, *Next*, *Next*, and then *Finish*

Figure 2-2 shows how the layout should look.

8. Hide *runoff.shp* and *drainage.shp* by deselecting their icons in the Project Explorer

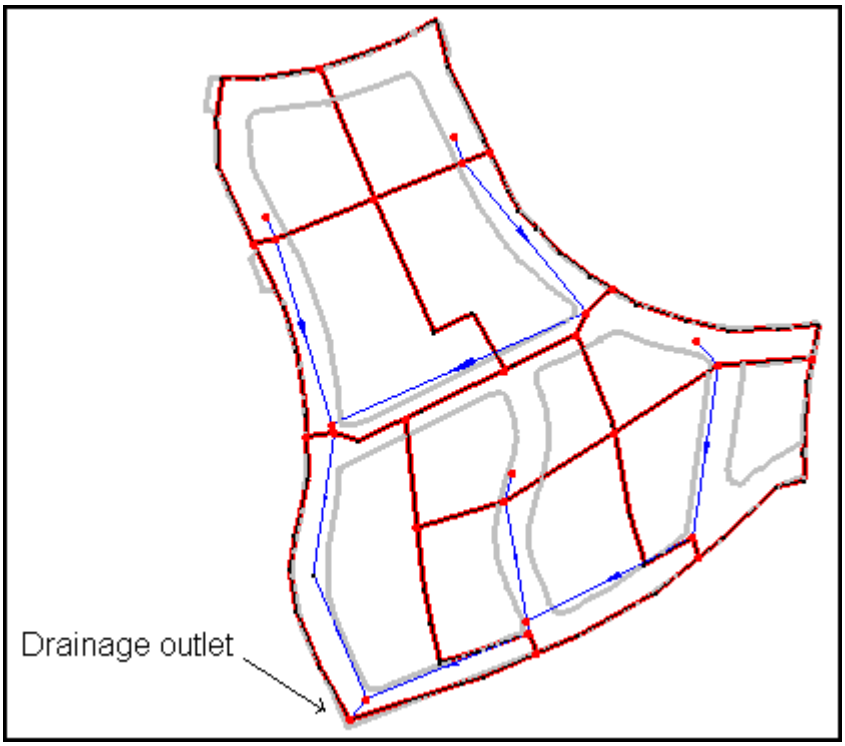


Figure 2-2: Drainage basins and streams for the subdivision.

Before defining nodes as storm drains/drainage outlets, we will hide the Runoff Coefficient coverage in order to simplify the screen display

9. Toggle the visibility check box for the *Runoff Coefficient* coverage off in the Project Explorer

10. Ensure that *Drainage* is the active coverage

11. Choose the *Select Feature Point/Node* tool

12. Double-click on the node labeled Drainage Outlet in Figure 2-2

13. Change the Point Type to *Drainage Outlet* and select *OK*

14. While holding the SHIFT key, select each of the nodes labeled as Storm Drains in Figure 2-3. These points correspond to manhole locations in the pipe network. Be sure to select the nodes that intersect with the basin boundary lines.

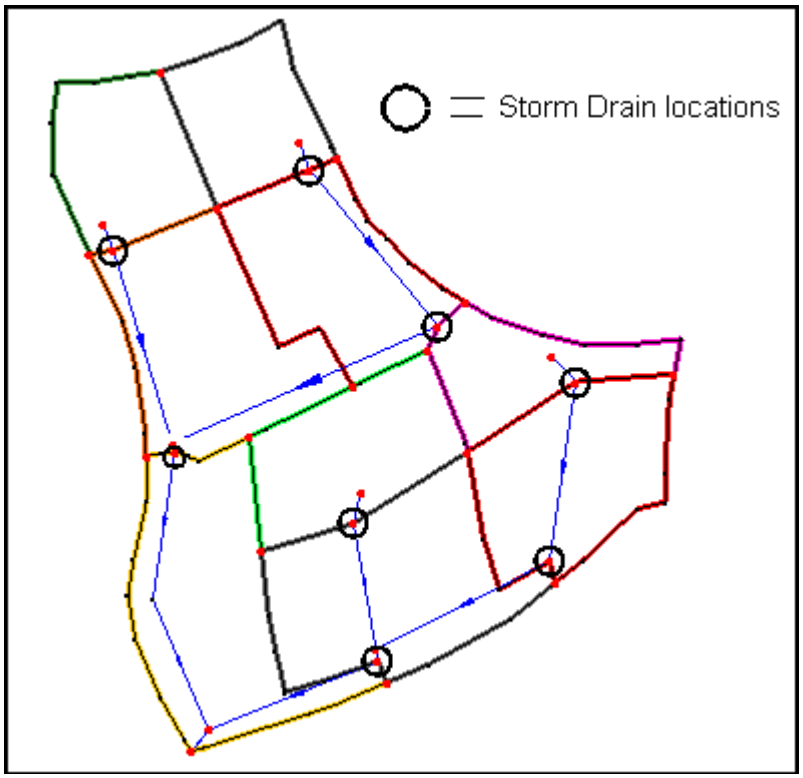


Figure 2-3: Storm Drain node locations.

15. Select ***Feature Objects | Attributes***
16. Change the Point type to *Storm drain inlet*
17. Click the *Properties* button
18. Change the Inlet Type to *Curb opening in sump condition*
19. Enter the dimensions shown in Table 2-1

Table 2-1: Values for the curb parameters.

Length	Width of Depression	Height	Inlet depression
4.0	3.0	1.0	0.25

20. Select *OK* twice to exit both dialogs
21. Select ***Feature Objects | Compute Basin Data***
22. Click the *Current Coordinates* button
23. Set the horizontal and vertical units as *U.S. Survey Feet*
24. Select *OK*

25. Set Basin Areas to *Acres* and Distances to *Feet*

26. Select *OK*

2.3.3 Computing Runoff Coefficients

Composite runoff coefficients must be computed for each drainage area. A weighted average for each drainage area is calculated from data in the Runoff Coefficient coverage. To compute the basin runoff coefficients:

1. Switch to the *Hydrologic Modeling* module

2. Select ***Calculators*** **|** ***Compute GIS Attributes***

3. Set the Computation type as *Runoff coefficients*

4. Select *OK*

Composite runoff coefficients for each drainage area are computed and displayed on the screen.

2.3.4 Entering Times of Concentration

The Time of Concentration for a basin can be calculated using a Time Computation. For this exercise, however, it is assumed that the TC values have already been calculated, and we will input them manually.

1. Change the Model drop-down box (located towards the top of the WMS interface) to *Rational*

2. Choose the *Select Basin* tool

3. Double-click on the basin icon for the basin in the lower left-hand corner of the subdivision

4. Enter **14** for the Time of Concentration (minutes)

5. Select each of the remaining drainage basins and assign Time of Concentration values using Figure 2-4 as a guide

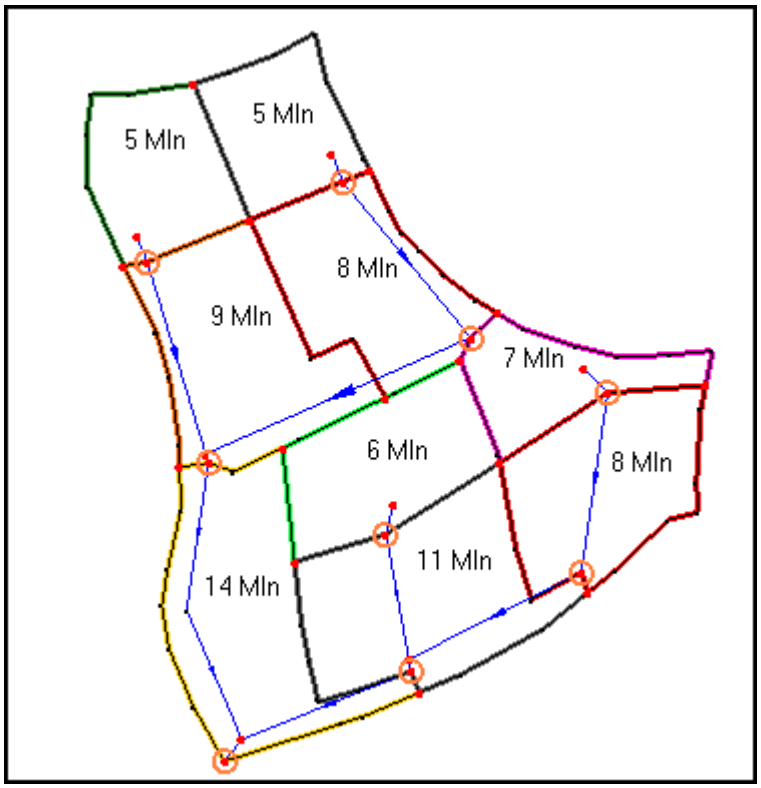


Figure 2-4: TC values for the drainage areas.

6. Select *OK* to close the Rational Method dialog

We have now set up a traditional rational simulation for the Cougar Estates subdivision. The only remaining step is to define the IDF curves. This step will be included as part of the Storm Drain process. We will now import a pre-defined pipe network, assign properties to the network, and link it to the Drainage coverage.

2.4 Importing the Pipe Network

As mentioned above, we will be importing a pipe network to use in our Storm Drain simulation. This network was defined as a shapefile, and will be converted to feature objects in WMS. WMS can also open DXF files and convert them to feature objects. Alternatively, users can manually create a pipe network using the Create Feature Arc tool.

1. Right-click on the Coverages folder in the Project Explorer
2. Select ***New Coverage***
3. Change the Coverage type to *Storm Drain-FHWA*
4. Select *OK*

5. Select ***File | Open***
6. Open *"pipenetwork.shp"*
7. Switch to the *GIS* module
8. Select ***Mapping | Shapes->Feature Objects***
9. Select *Yes* to use all visible shapes
10. Select *Next*, *Next*, and *Finish*
11. Hide *pipenetwork.shp*

2.4.1 Entering Job Control Data

1. Switch to the Map module
2. Select ***Storm Drain | Job Control***
3. Enter *"Storm Drain Rational"* for the Title
4. Choose the *English* option for the Units
5. Click on the *Pipe Data Properties* button

This dialog allows users to define a set of pipe properties that will be applied to all pipes in the network. In addition to assigning pipe properties on a global scale, users can define specific properties to individual pipes, or to a group of pipes by selecting them with the Select Feature Arc tool and selecting the Attributes command from the Feature Objects menu. For this exercise, we will simply define global pipe properties.

6. Change the Pipe type to *Concrete-Smooth*

When a user specifies a pipe type, WMS fills in an appropriate Manning's n roughness value based on that pipe type. Users can also enter/edit the Manning's n value directly in this dialog. Users can change the Manning's n values that are assigned to the various Pipe Types. These values can be accessed and edited by opening the Display Options dialog, choosing Map Data, and changing the Active Coverage field to Storm Drain.

7. Enter the values displayed in Figure 2-5

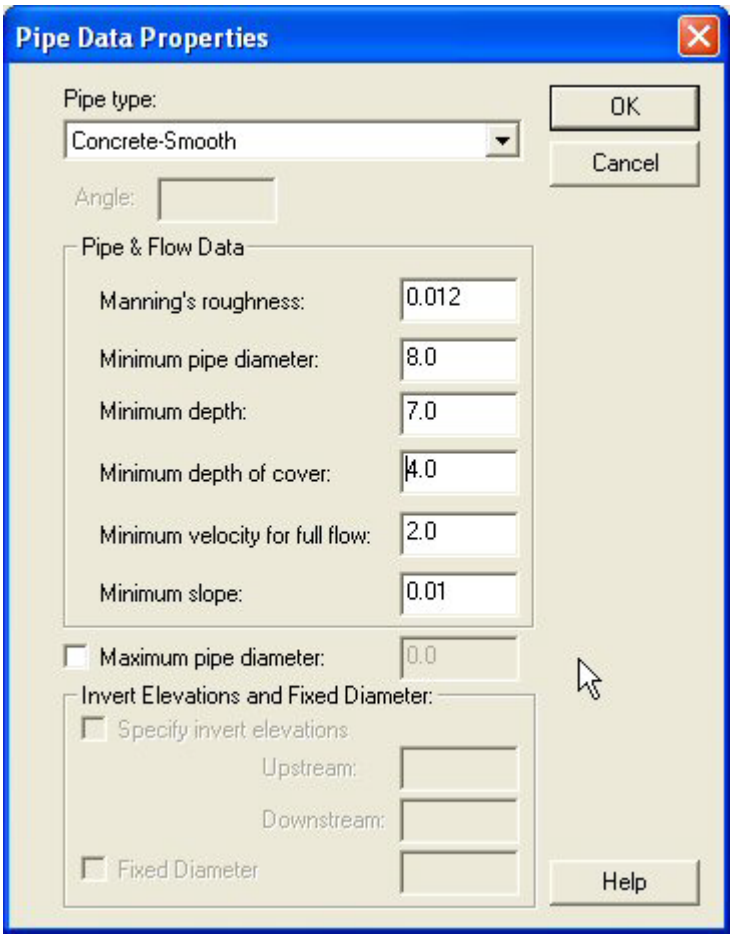


Figure 2-5: Pipe property values for the pipe network.

8. Select *OK* to exit the Pipe Data Properties dialog

2.4.2 Defining Rainfall Data

1. Click on the *Rainfall Data* button
2. Select the *Run Compute IDF Curve Dialog* option
3. Select *OK*
4. Choose the *User Supplied Data* option as the IDF curve computation
5. Click the *Define Storm Data* button
6. Change the Recurrence value to **25** yr
7. Enter the precipitation values shown in Figure 2-6

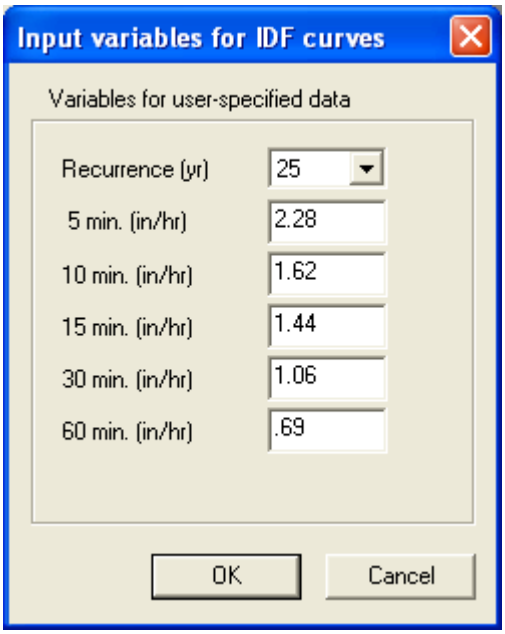


Figure 2-6: Values for computing the 25 yr IDF curve.

8. Select *OK*

9. Highlight the line corresponding to the 25-yr precipitation values from the window in the upper right-hand corner of the dialog

10. Select *Done*

11. Select *OK* twice to return to the WMS interface

12. Switch to the *Hydrologic Modeling* module

13. Choose the *Select Basin* tool

14. Double-click on the basin labeled *4B*

Notice that the intensity value computed in the Storm Drain interface has been copied here as well, and Q is now computed for the basin.

15. Select *OK* to exit the Rational Method dialog

2.4.3 Defining the Outfall and Manhole Locations

1. Click on the *Storm Drain* coverage in the Project Explorer to select it

2. Choose the *Select Feature Point/Node* tool

3. Double-click on the node in the lower left-hand corner of the subdivision as shown in Figure 2-7

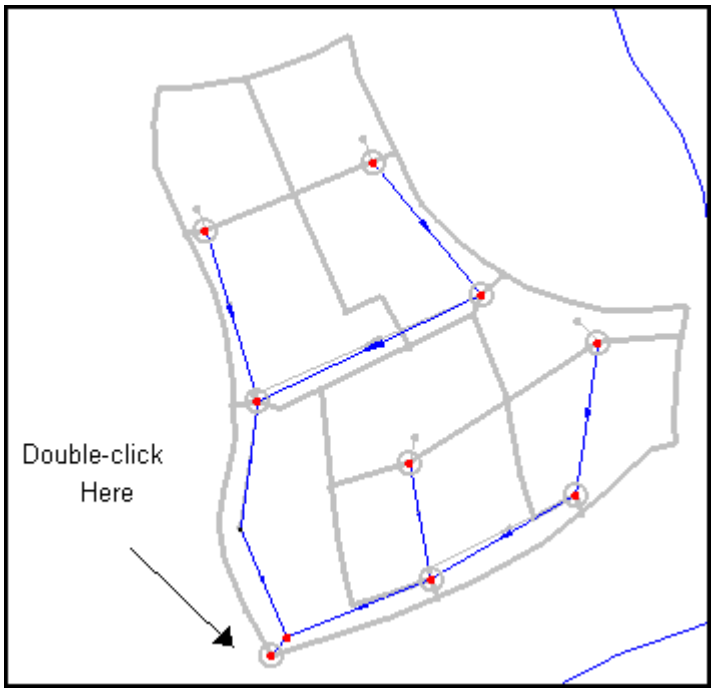


Figure 2-7: Outfall location.

4. Change the node type to *Outfall*

5. Select *OK*

6. Multi-select the nodes indicated in Figure 2-8 as Mahhole Locations by holding down the SHIFT key as you select each node

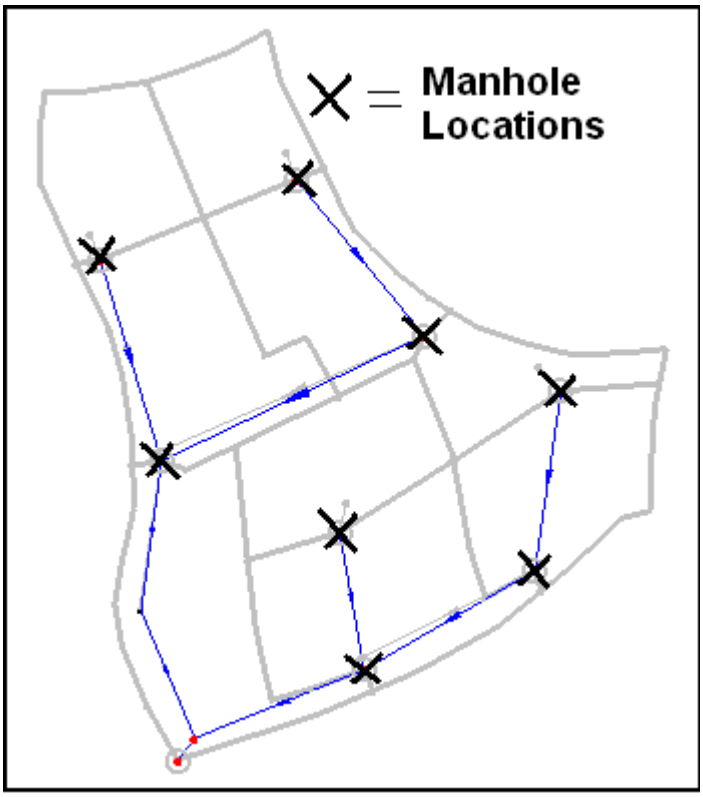


Figure 2-8: Manhole locations.

7. Select ***Feature Objects | Attributes***
8. Change the node type to *Access Hole*
9. Enter **4.0** as the Access hole width
10. Change the Bench Type to *Flat Bench*
11. Select *OK*

The Pipe Drain Network is now defined

2.5 Linking Nodes and Assigning Elevations

To properly portray the relationship between the Drainage coverage and the Storm Drain coverage, we need to explicitly define links between their corresponding nodes. We also need to assign elevations to all linked nodes. Elevations can either be defined manually, or with the use of a background TIN or DEM. We will use the TIN that we opened earlier in this exercise for assigning elevations.

1. Select ***Storm Drain-FHWA | Link Nodes***

2. Click the *Auto Link* button. This will link the storm drain inlets from the *Drainage* coverage to the manholes in the *Storm Drain* coverage. All nodes should be linked except the one node just upstream of the outfall. This node is a pipe junction and is not associated with a storm drain inlet.

3. Select *OK*

4. Select ***Storm Drain-FHWA | Assign Pipe Elevations***. This command automatically assigns elevations to nodes based on an underlying TIN or DEM, if one is present.

With Drainage and Storm Drain nodes linked together and elevations assigned to our pipe network, we are ready to save the Storm Drain model.

2.6 Saving the Simulation and Running Storm Drain

1. Select ***File | Save As***

2. Make sure the Save as type filter is set to *WMS XMDF Project Files (*.wms)*

3. Enter *"wms_storm_rat"* for the File name

4. Select *Save*

5. Select ***Storm Drain-FHWA | Save Simulation***

6. Save the file as *"wms_st_r"*

7. Select ***Storm Drain-FHWA | Run Simulation***

8. Verify that the input file name is *wms_st_r.hda*

9. Verify the output file name is *wms_st_r.lst*

The input and output file name prefixes can be up to 8 characters in length. If you enter an input file name longer than 8 characters, Storm Drain will not run. If you enter an output file name longer than 8 characters, it is truncated to 8 characters when Storm Drain writes the file. Also all directory names in the path of the input/output files cannot contain more than 8 characters.

10. Select *OK*

11. Once Storm Drain (HYDRA) has finished running, select *Close*

12. Select *OK*

13. Select ***File | Edit File***

14. Open *"wms_st_r.lst"*

15. Select *OK* to open the file with Notepad

By browsing through this file, you will see all of the parameters calculated by the Storm Drain (HYDRA) model, such as recommended pipe diameters, pipe invert elevations, flows, velocities, and hydraulic grade line computations.

16. Close Notepad

CHAPTER 3

Storm Drain: Hydrographic Design

Storm Drain is a hydraulic analysis and design program for storm drain and sanitary sewer systems. It was developed by the Federal Highway Administration (FHWA) with the intent of providing hydraulic engineers a quick and accurate method of designing and analyzing storm drain, sanitary sewer, and combination systems. The Storm Drain interface in WMS uses the same HYDRA program to perform calculations that is currently part of the FHWA's HYDRAIN suite of hydraulic programs.

This exercise demonstrates how to use WMS to set up a Hydrographic Drainage simulation for use with Storm Drain. We will set up drainage areas and Runoff Coefficients for a subdivision, similar to the previous chapter, but we will also specify gutter locations in the Drainage coverage. As illustrated in this exercise, the pipe network does not need to flow parallel or in the same direction as the over-lying drainage. However, there are some limitations in the way the drainage coverage is constructed in order to run a hydrographic analysis with Storm Drain. The most important limitation is that each inflow hydrograph needs to be associated with one and only one gutter at the storm drain inlet. In other words "extra" stream arcs that are not part of the gutter leading into the inlet should be deleted.

In this analysis we will generate input hydrographs using the Rational Method calculator, but it should be emphasized that any hydrologic method that produces a hydrograph can be used, or hydrographs may be entered manually.

3.1 Objectives

In this exercise, we will set up a hydrographic storm drain simulation for a hypothetical subdivision. The objective of this exercise is to teach you the basic steps for defining a Storm Drain input file for hydrographic analysis. These steps include the following:

1. Build the drainage basins and make adjustments for limitations in doing a hydrographic analysis
2. Perform a simple Rational Method analysis to generate input hydrographs
3. Digitize a small pipe network based on the surface drainage
4. Define the necessary parameters for the pipe network
5. Run Storm Drain (HYDRA)
6. View the solution

3.2 Developing the Surface Drainage Coverage

We will begin with a drainage coverage created with techniques typical to those described in earlier chapters. In order to prepare the drainage coverage for storm drain analysis we will eliminate all of the stream arcs, except those that lead into a given outlet (a storm drain inlet in this case). Having extra branching streams prevents WMS from building the storm drain input file correctly. You must decide which arc represents the primary stream (in this case a gutter) into an inlet and delete the others.

1. Close all instances of WMS
2. Open WMS
3. Switch to the *Map* module
4. Select ***File*** **|** ***Open***
5. Locate the folder ***C:\Program Files\WMS84\tutorial\stormhydro***
6. Open *"sdstart.map"*

3.2.1 Eliminating Unnecessary Stream Arcs and Nodes

Figure 3-1 shows the drainage coverage with the extra streams highlighted (arrows provide additional clarification). In order to prepare the drainage coverage correctly you will need to delete these stream arcs.

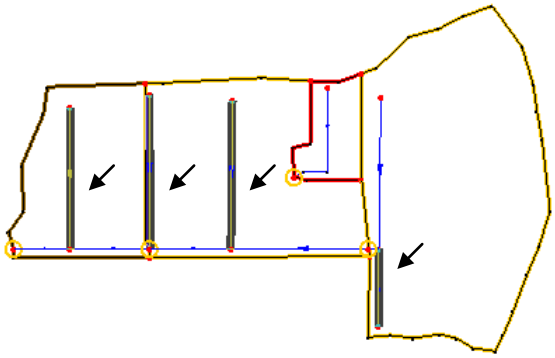


Figure 3-1: Initial drainage coverage with unnecessary stream arcs shown.

1. Select the *Select Feature Arc* tool
2. While holding down the SHIFT key (for multi-selection) select the four arcs highlighted in Figure 3-1
3. Select the *Delete* macro (or ***Edit*** **|** ***Delete*** or the DELETE key)
4. Select *OK*

You must now delete the extra nodes that were at the intersection of the arcs just deleted and the remaining arcs so that there is just a single arc entering the storm drain inlet points.

5. Select the *Select Feature Point/Node* tool
6. While holding down the SHIFT key, select the three nodes identified by arrows in Figure 3-2 (be sure to select the point to the right of the outlet at the far right and not the point that is the outlet)

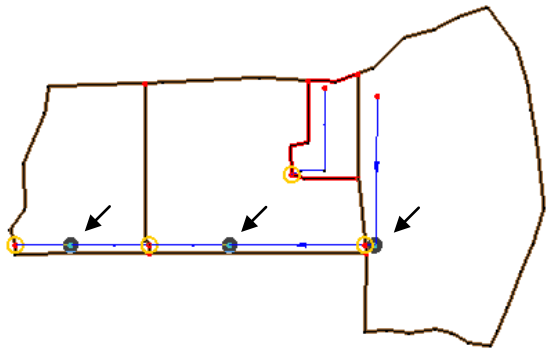


Figure 3-2: Nodes that need to be converted to vertices.

7. Select ***Feature Objects | Vertex*** <-> ***Node***

3.2.2 Initializing Storm Drain Simulation

In order to assign the remaining stream arcs to be gutters and the outlet points as storm drain inlets we must initialize a storm drain analysis.

1. Select *Storm Drain-FHWA* from the Models drop-down list at the top of the screen

2. Select ***Storm Drain-FHWA | New Simulation***

3.2.3 Assigning Stream Arcs as Gutters

1. Select the *Select Feature Arc* tool

2. While holding down the SHIFT key, select all four stream arcs as indicated in Figure 3-3

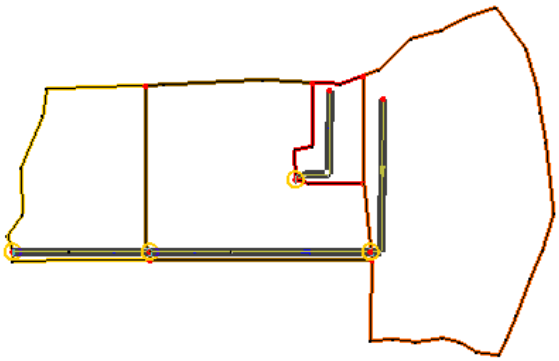


Figure 3-3: Arcs to be selected.

3. Select ***Feature Objects | Attributes***

4. Select *Gutter* for the Feature Arc Type

5. Choose the *Atts* button to define the properties of the gutters

6. Set the type to *Uniform Gutter* (this should be the default) and set the Manning's n to **0.015**, the Roadway Cross-slope to **0.02**, and the width to **2.0**

7. Select *OK*

8. Select *OK*

3.2.4 Assigning Outlets as Storm Drain Inlets

For this analysis, two of the outlets will be defined in areas of depression (sump conditions) while the other two will have bypass flow (on grade conditions).

1. Select the *Select Feature Point/Node* tool

2. While holding down the SHIFT key select both of the outlet points of the entire basin system, and the outlet of the isolated interior basin as indicated in Figure 3-4

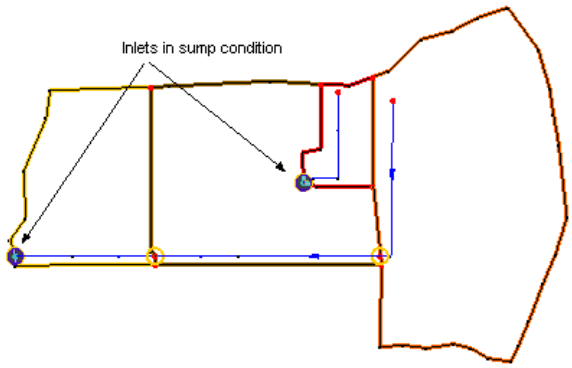


Figure 3-4: Outlet points to be selected.

3. Select ***Feature Objects | Attributes***

4. Set the Drainage Feature Point Type to be *Storm drain inlet*

5. Choose the *Properties* button

6. Set the Type to be *Grate in sump condition*, the Subtype to be *Reticuline*, the Width to be **2.0**, the Length to be **6.0**, the Inlet Perimeter to be **16.0**, and the Area to be **8.5**

7. Select *OK*

8. Select *OK*

9. Select one of the other two remaining outlets (this will ensure that the two selected outlets become unselected)

10. Now while holding down the SHIFT key select the remaining outlet (the one that has not yet been selected)

11. Select ***Feature Objects | Attributes***

12. Set the Drainage Feature Point Type to be *Storm drain inlet*

13. Choose the *Properties* button

14. Set the Type to be *Grate on grade*, the Subtype to be *Reticuline*, the Width to be **2.0**, and the Length to be **6.0**

15. Select *OK*

16. Select *OK*

17. Select ***Feature Objects | Compute Basin Data***

18. Select *OK*

3.3 Running a Rational Analysis

We will perform a simple Rational analysis of this subdivision and use the Rational method hydrographs as input for the hydrographic storm drain analysis. You should remember, though, that any of the hydrologic analysis models in WMS can be used, or input hydrographs may be entered manually.

1. Select the *Hydrologic Modeling* module

2. Change the Model drop-down box located at the top of the WMS interface to *Rational*

3.3.1 Defining Rational Method Basin Parameters

1. Double-click on one of the basin icons using the *Select Basin* tool and define the basin parameters according to Figure 3-5 and Table 3-1. Select another basin and define its parameters until all basin data is defined. For more details on defining a Rational Simulation see the chapter on Rational method analysis (Volume 2, Chapter 5).

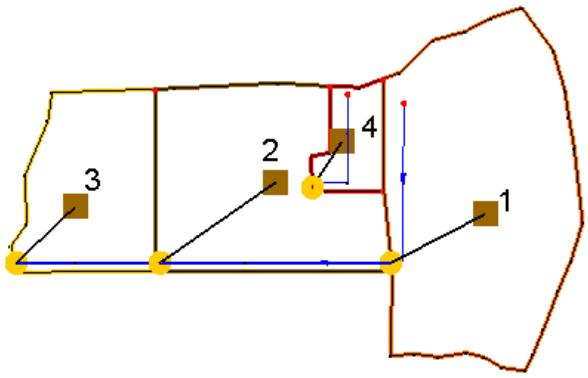


Figure 3-5: Basin locations.

Table 3-1: Rational Method basin parameters.

Basin	C	I (in/hr)	Tc (min)
1	0.10	2.4	22
2	0.15	2.7	18
3	0.15	3.1	12
4	0.20	3.8	07

2. Select *OK* in the Rational Method dialog

3.3.2 Defining Rational Method Routing Parameters

1. Define the intensity and routing time for the four confluence points connected to the basins according to Table 3-2. As with basins, reach data are entered by double-clicking on the yellow outlet icons using the *Select Outlet* tool .

Table 3-2: Rational Method routing parameters.

Outlet	Routing Lag (min)	I (in/hr)
1	5	2.4
2	4	1.9
3	0	1.7
4	0	3.8

2. Select *OK* in the Rational Method dialog

3.3.3 Computing Hydrographs

1. Double-click on the outlet for the isolated interior basin (the outlet of basin 4 in Figure 3-5

2. Choose the *Compute Hydrographs* button at the bottom of the Outlet column

3. Select *Done* (use the default settings for the Rational method hydrograph)

4. Select *OK*

5. Double-click on the outlet for the entire watershed (the outlet of basin 3 in Figure 3-5)

6. Choose the *Compute Hydrographs* button at the bottom of the Outlet column

7. Select *Done* (use the default settings for the Rational method hydrograph)

8. Select *OK*

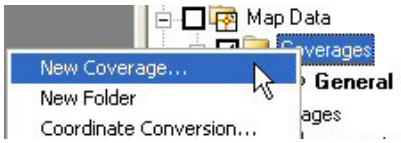
At this point you should have hydrographs defined for the drainage coverage. These hydrographs will be used as input to the storm drain problem.

3.4 Creating the Pipe Network

You will now create a storm drain pipe network that connects the isolated interior basin to the others and exits west of the watershed outlet. You will do this by creating a Storm Drain coverage and then digitizing the storm drain pipes using the *Create Feature Arc* tool.

3.4.1 Digitizing Storm Drain Pipes

1. Right-click on the Coverages folder in the Project Explorer



2. Select ***New Coverage***

3. Change the Coverage type to *Storm Drain-FHWA*

4. Select *OK*

5. Select the *Create Feature Arc* tool

6. Beginning at a point just to the left (west) of the outlet of the watershed (basin 3) begin a pipe network segment and end it by double-clicking on top of the outlet of basin 3.

7. Now complete the storm drain network by creating the pipe segments from downstream to upstream between basin 3 and basin 2, basin 2 and basin 1, and a connection from the pipe between basin 2 and basin 1 to the outlet of basin 4. Be sure that feature nodes (segment endings) are created by double-clicking to end the arc to correspond with the storm

drain inlets of the drainage coverage. Your storm drain coverage should look similar to Figure 3-6.

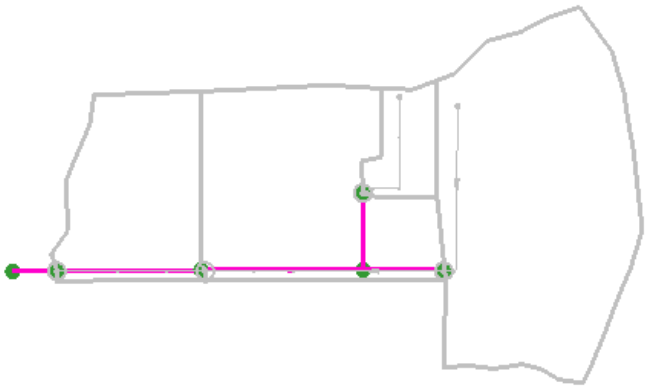


Figure 3-6: View of the Storm Drain coverage.

3.4.2 Entering Storm Drain Parameters

1. Select ***Storm Drain-FHWA | Job Control***
2. Set the title to: *Hydrographic Storm Drain Analysis*
3. Set the Criteria Switch to *3 - Storm (Hydrographic) only*
4. Set the timestep to **2**
5. Set the Units to *English*
6. Choose the *Pipe Data Properties* button
7. Set the values according to Figure 3-7

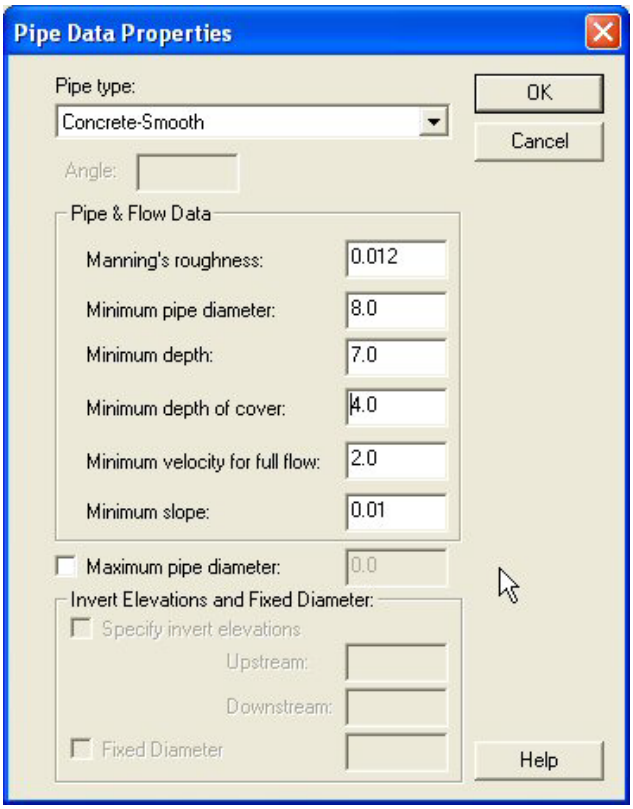


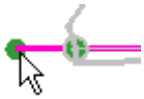
Figure 3-7: Pipe properties.

8. Select *OK*

9. Select *OK*

3.4.3 Defining the Outfall and Manhole Locations

1. Choose the *Select Feature Point/Node* tool



2. Double-click on the node to the left of the drainage basin

3. Set the Node Type as *Outfall*

4. Select *OK*

5. Use the SHIFT key to select the four nodes of the storm drain coverage that correspond to the four storm drain inlets (manholes) of the drainage coverage as shown in Figure 3-8

6. Select ***Feature Objects | Attributes***

7. Set the Node Type as *Access Hole*

8. Set the width to be **4.0**

9. Select *OK*

You could define each manhole separately by double-clicking on the node and setting the node type and manhole width, but it is more efficient to define them all at once. At this point you should have one outfall node, four manhole nodes, and a single pipe junction in your storm drain coverage as identified in Figure 3-8. If this is not the way your storm drain coverage is set up, then correct it now.

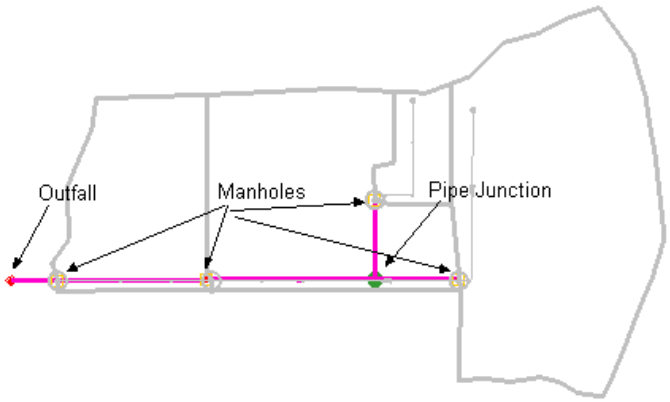


Figure 3-8: The correct setup of the Storm Drain coverage.

3.4.4 Linking Nodes

To properly portray the relationship between the drainage coverage and the storm drain coverage, we need to explicitly define links between corresponding nodes.

1. Select ***Storm Drain-FHWA | Link Nodes***

2. Change the tolerance to **25**

3. Click the *Auto Link* button. This should link the four storm drain inlet nodes of the drainage coverage to the four manholes of the storm drain coverage. If not, increase the tolerance by an increment of 5 and select the *Auto Link* button until the four node pairs are linked.

4. Select *OK*

3.4.5 Assigning Elevations and Mapping Hydrographs

We also need to assign elevations to all the nodes in the Storm Drain and Drainage coverages. Elevations can either be defined manually, or with the use of a background TIN or DEM. In this exercise we will manually define them.

1. Toggle the display of the *Drainage* coverage off in the Project Explorer

2. Choose the *Select Feature Point/Node* tool

3. Set the elevations of the nodes in the Storm Drain coverage according to the values given in Figure 3-9. To set an elevation, select the node and change the elevation in the Feature Point Z edit field in the Properties window on the right of the WMS interface and press Enter.

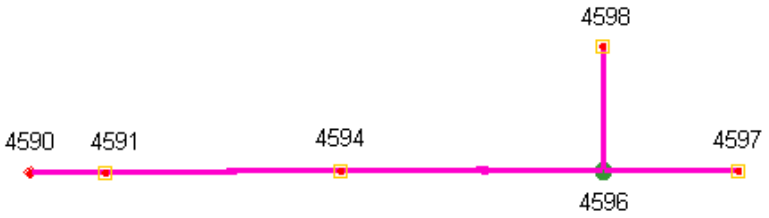


Figure 3-9: Node elevations for the storm drain pipes.

4. Toggle the *Drainage* coverage on and the *Storm Drain-FHWA* coverage off in the Project Explorer

5. Select the *Drainage* coverage in the Project Explorer so that it is the active coverage

6. Select and set the elevations of all the nodes connected to the gutter arcs according to Figure 3-10

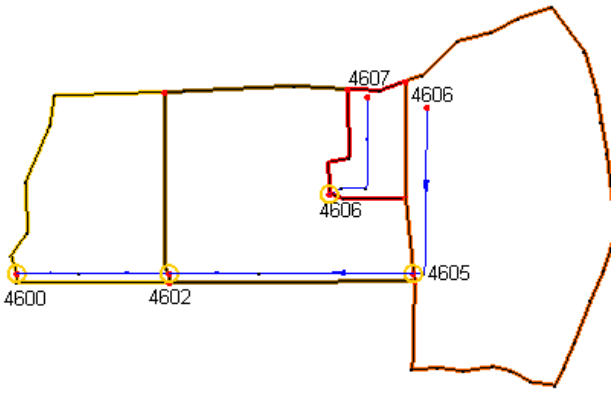


Figure 3-10: Node elevations for the gutters.

7. Set *Storm Drain-FHWA* as the active coverage in the Project Explorer

8. Select ***Storm Drain-FHWA | Map Hydrographs***

With Drainage and Storm Drain nodes linked, and elevations assigned to our pipes and gutters, we are ready to save and run the Storm Drain model.

3.5 Saving the Simulation and Running Storm Drain

1. Select ***File | Save As***
2. Make sure the Save as type filter is set to *WMS XMDF Project Files (*.wms)*
3. Enter *"wms_storm_hyd"* as the File name
4. Select Save
5. Select ***Storm Drain-FHWA | Save Simulation***
6. Save the file as *"wms_st_h"*
7. Select ***Storm Drain-FHWA | Run Simulation***
8. Set the input file name to *"wms_st_h.hda"* and the output file name to *"wms_st_h.lst"*

The input and output file name prefixes can be up to 8 characters in length. If you enter a name longer than 8 characters, it is truncated to 8 characters when Storm Drain writes the file. Further, no directory in the path to the files can be longer than 8 characters.

9. Select *OK*
10. Once Storm Drain (HYDRA) has finished running, select *Close*
11. Select *OK*
12. Select ***File | Edit File***
13. Open *"wms_st_h.lst"*
14. Select *OK* to open the file with Notepad

By browsing through this file, you will see all of the parameters calculated by the Storm Drain model, such as recommended pipe diameters, pipe invert elevations, flows, velocities, and hydraulic grade line computations.

15. Close Notepad